INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS
Int. J. Numer. Meth. Fluids 2000; **32**: 569–584

Semi-structured meshes for axial turbomachinery blades

L. Sbardella, A. I. Sayma and M. Imregun*

Imperial College of Science, Technology and Medicine, Mechanical Engineering Department, Exhibition Road, London SW7 2BX, U.K.

SUMMARY

This paper describes the development and application of a novel mesh generator for the flow analysis of turbomachinery blades. The proposed method uses a combination of structured and unstructured meshes, the former in the radial direction and the latter in the axial and tangential directions, in order to exploit the fact that blade-like structures are not strongly three-dimensional since the radial variation is usually small. The proposed semi-structured mesh formulation was found to have a number of advantages over its structured counterparts. There is a significant improvement in the smoothness of the grid spacing and also in capturing particular aspects of the blade passage geometry. It was also found that the leading- and trailing-edge regions could be discretized without generating superfluous points in the far field, and that further refinements of the mesh to capture wake and shock effects were relatively easy to implement. The capability of the method is demonstrated in the case of a transonic fan blade for which the steady state flow is predicted using both structured and semi-structured meshes. A totally unstructured mesh is also generated for the same geometry to illustrate the disadvantages of using such an approach for turbomachinery blades. Copyright © 2000 John Wiley & Sons, Ltd.

KEY WORDS: quasi-conformal mapping; semi-structured mesh formation; turbomachinery blades

1. INTRODUCTION

It is well known that the grid structure must be selected carefully in order to achieve an accurate resolution of complex flow fields typical of axial-flow turbomachines. The minimization of skewness and the optimization of smoothness generally result in a faster convergence as well as less solution dependence on the grid density, therefore reducing computational cost both in terms of memory and CPU time. As a consequence, the grid generation procedure should be considered an integral part of the numerical method.

When performing a numerical simulation of turbulent–viscous flow in turbomachinery passage, the following aspects are of importance: (1) accurate leading- and trailing-edge flow descriptions, (2) wake resolution, (3) proper gridding in the throat area where most of the shock is expected to occur and (4) imposition of periodicity.

* Correspondence to: Mechanical Engineering Department, Imperial College of Science, Technology and Medicine, Exhibition Road, London SW7 2BX, U.K.

CCC 0271–2091/2000/050569–16$17.50

Received January 1998
Revised July 1998

Historically, mesh generation techniques for turbomachinery blades use structured hexahedral representations, the most commonly used ones being H-type, C-type, and O-type. These meshes are obtained either by using an algebraic approach or by solving a system of elliptic partial differential equations [1–3]. H-type meshes have been by far the most common choice in turbomachinery applications as they are very easy to generate, the imposition of periodicity is straightforward and the mesh density before, inside, and after the blade passage can be easily controlled. However, the leading- and trailing-edge descriptions are poor and a large amount of superfluous points are generated in the region between the inflow and the leading-edge. O-type grids are not very effective in capturing the wake and their quality outside the passage is very poor. This can smear the bow shock away from the leading edge of a transonic compressor or the outgoing shock of a transonic turbine blade. On the other hand, C-type meshes can capture the wake structure if they are carefully generated but their quality in the region between the inflow and the leading-edge is not suitable to resolve bow shock accurately.

A different approach is to use unstructured triangular meshes for two-dimensional turbomachinery calculations. Fully unstructured grids offer good flexibility and most of the flow features can be captured with good accuracy via mesh refinement. Three-dimensional unstructured meshes are widely used in external aerodynamics but they have rarely been applied to turbomachinery cases. The main difficulty associated with such meshes is their isotropic nature. The very fact that tetrahedral unstructured meshes do not exhibit any preferred direction is what makes them ideal for discretizing arbitrarily complex configurations. In fact, most of the unstructured mesh generation techniques rely on this property. However, when a configuration with a preferred direction, such as a turbomachinery blade, is to be discretized, and different resolutions are desired in the various directions, unstructured mesh generation techniques are known to experience great difficulties in meeting such requirements. Turbomachine blades require high resolution near their leading and trailing edges and radial spacing can be relatively coarse. When using an isotropic unstructured mesh, the high leading and trailing edge resolution requirements also result in a high radial resolution in these areas, a feature that greatly increases the number of grid points. Such a degree of radial resolution is superfluous, since the radial gradients are known to be relatively small for turbomachinery blade flows. Similar difficulties occur in the boundary layer regions near a wall for high-Reynolds number viscous flows, where the normal gradients are several orders of magnitude greater than the streamwise gradients.

The considerations above lead to the use of semi-structured meshes for turbomachinery blades. The aim of this paper is to present a novel approach for discretizing turbomachinery blades by using a combination of structured and unstructured meshes, the former in the radial direction and the latter in the axial and tangential directions. The basic idea relies on the fact that blade-like structures are not strongly three-dimensional since the radial variation is usually small. It is therefore possible to start with a structured and body fitted two-dimensional O-grid around a given aerofoil section to resolve the boundary layer. This core mesh is then extended in an unstructured fashion up to the far-field boundaries, the triangulation being performed using an advancing front technique [4,5]. Once this two-dimensional grid is generated, it is projected to the remaining radial sections via quasi-conformal mapping techniques. When all such radial sections are formed, a three-dimensional prismatic grid is obtained by simply connecting the corresponding points of different layers. In this way, hexahedral elements are generated in the viscous region and triangular prisms in the rest of the solution domain. Again, such an approach presents a distinct advantage over totally unstructured grids, which are usually confined to tetrahedral elements only, though this limitation is usually a consequence of the unavailability of general viscous mesh generators.

2. GENERATION OF PRISMATIC GRID

Since this three-dimensional mesh-generation procedure involves several distinct stages, like geometry modeling, two-dimensional hybrid mesh generation, mapping etc., this section will first describe the methodology without undue details. The next section will focus on the proposed quasi-conformal mapping procedure which links the two-dimensional radial section meshes together. The method is based on five main stages:

- Mapping procedure to project all radial levels of the blade into two-dimensional planes, using local co-ordinates.
- Generation of a two-dimensional hybrid mesh for a given radial section.
- Generation of a coarse body fitted structured mesh for all radial sections.
- Inverse mapping of the unstructured mesh into a structured one at the same radial level.
- Direct mapping to obtain the final unstructured mesh at all radial levels.

2.1. Geometry modelling

Turbomachinery blade geometry is usually defined at a number of radial sections. In the general case, these radial sections will lie on three-dimensional surfaces $\sigma_i(x, r, \theta)$, where i indicates the section index. Using parametric co-ordinates u and v, a typical surface can be defined as:

$$\begin{cases} x_\sigma = x(u, v) \\ r_\sigma = r(u, v) \\ \theta_\sigma = \theta(u, v) \end{cases} \tag{1}$$

The starting point of the present method is the projection of the radial sections into parametric two-dimensional planes, using the local co-ordinate system u and v. In this way the mesh-generation procedure will deal with the plane section only, thus the geometric dimension is reduced from three to two.

2.1.1. Unstructured mesh generation. Once all radial sections are mapped onto two-dimensional planes, a hybrid quadrilateral–triangular mesh is generated in a two-dimensional plane, which corresponds to a certain radial level (usually the middle one). The quadrilateral part of the mesh takes the form of a body-fitted O-grid, which is generated using a system of elliptic partial-differential equations. The orthogonality of this structured mesh is very important for an accurate resolution of the turbulent boundary layer that originates from high-Reynolds number flows.

The remaining part of the domain is discretized using an unstructured mesh generator which uses an advancing front algorithm [4,5]. A distinctive feature of this method is that triangles and points are generated simultaneously. Such an approach enables the generation of elements with variable sizes and stretching, and hence, differs from Delaunay-type mesh generators [5,6].

2.1.2. Quasi-conformal mapping. An important part of the mesh-generation strategy is the mapping procedure to project the unstructured mesh, generated at a given radial level, to all radial blade-definition levels. A necessary condition for a mapping function is that it must associate a given point of the first plane with one, and only one, point of the second plane. Moreover, a mapping function should also guarantee that a given angle in one plane is mapped into a similar-valued angle in the target plane (quasi-conformal mapping). This last property is essential in order to minimize the skewness of the mesh, especially for highly twisted fans. As will be illustrated in Section 3, the quasi-conformal mapping procedure will make use of structured meshes.

Once the unstructured mesh has been mapped to all radial blade surfaces, a prismatic mesh is obtained by simply connecting the corresponding points at consecutive levels. Moreover, in order to enhance the quality of the three-dimensional mesh, a smoothing procedure is performed. This operation alters the positions of the interior nodes without changing the topology of the mesh. The element sides are considered as springs of stiffness proportional to the length of the side. The nodes are moved until the spring system is in equilibrium, the position of which is found by Jacobi iterations.

3. QUASI-CONFORMAL MAPPING

The starting point for the quasi-conformal mapping is the generation of coarse structured quadrilateral grids for all radial sections. Following the approach of Steger and Sorenson [2], these meshes are obtained by solving a system of elliptic partial-differential equations. An essential requirement for such structured meshes is that they must be generated in exactly the same manner, i.e. with the same number of points and quadrilaterals. The mapping procedure is implemented as follows.

- **Geometry searching**. Each point J of the unstructured two-dimensional mesh must be located on quadrilateral E of the structured mesh.
- **Inverse mapping**. The Cartesian co-ordinates: $\vec{x}_J = (x_J, y_J)$ of point J, associated with the quadrilateral E are given by

$$\vec{x}_J = \sum_{I=1}^{4} \vec{x}_I N_I(\xi_J, \eta_J) \tag{2}$$

where (ξ_J, η_J) represent the local co-ordinates, $\vec{x}_I$ represents the Cartesian co-ordinates of nodes $I = 1, \ldots, 4$ of the quadrilateral E and N_I is the standard finite element bilinear shape functions, which take the form [7]

$$N_{\mathrm{I}} = \frac{1}{4}\begin{cases}(1-\xi)(1-\eta)\\(1+\xi)(1-\eta)\\(1+\xi)(1+\eta)\\(1-\xi)(1+\eta)\end{cases} \tag{3}$$

A Newton–Raphson method is used in order to obtain the values of ξ_J and η_J.

- **Direct mapping**. Once all points J of the unstructured mesh are associated with quadrilateral E and the local co-ordinates (ξ_J, η_J) are determined, the co-ordinates $\vec{x}_J$ of the points on the remaining radial sections are obtained directly using Equation (2).

The above-described steps are shown in Figure 1. The mapping method becomes fully conformal if a given element of the two-dimensional hybrid mesh lies within a single quadrilateral of the structured grid. In addition, the angles of this quadrilateral must remain the same for all radial sections [7]. If the above two conditions are not satisfied, the conformal property is not guaranteed and for this reason the procedure here has been labelled quasi-conformal. Finally, it is worth noting that the algorithm is not CPU-intensive, since the geometry searching and the inverse mapping are only performed once.

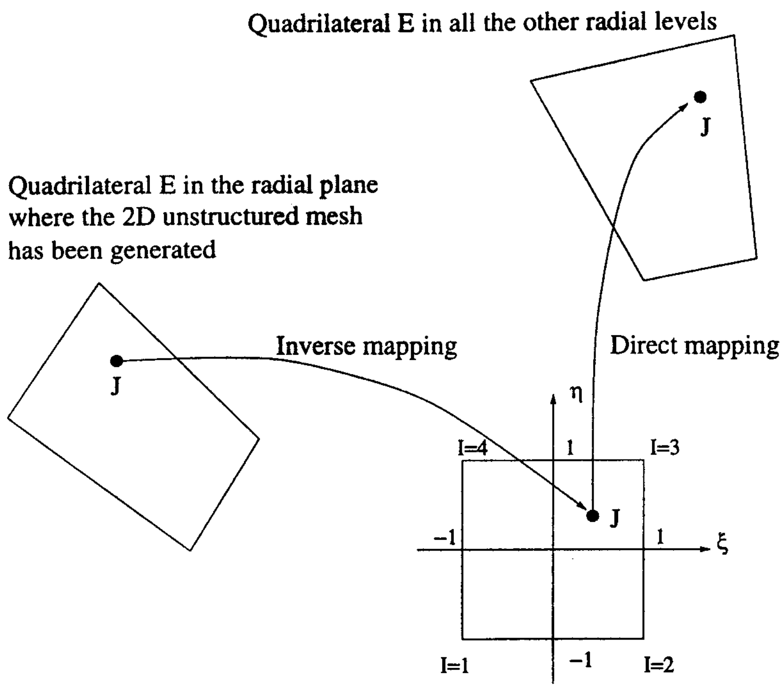


Figure 1. Mapping procedure.

4. EXAMPLES

A semi-structured mesh, generated with the proposed method, will now be used to calculate the steady flow field of a transonic fan blade at 70 per cent of design speed. The difficulty in obtaining satisfactory part-speed flow solutions is well known in the turbomachinery computational fluid dynamics (CFD) community and this is why such a solution has been attempted here. The Reynolds averaged compressible Navier–Stokes equations, together with the Baldwin and Barth turbulence model [8], are cast in terms of absolute velocity but solved in a relative non-Newtonian reference frame, rotating along with the blade, about the x-axis, with angular velocity Ω. The flow solver is an implicit, upwind-differencing algorithm in which the inviscid fluxes are obtained on the faces of each control volume using the flux difference splitting of Roe [9]. Second-order accuracy is obtained using the gradient information of the unknown variables at each control volume. In order to guarantee monotonicity of the scheme and not to deteriorate convergence to steady state, the modified van Leer pressure-based switch of Reference [10] is used. The viscous terms are evaluated with a finite volume formulation which is equivalent to a Galerkin-type approximation and which results in a central-difference-type formulation of the viscous losses. The solution at each time step is updated using a linearized backward-Euler, time differencing scheme. The linear system of equations is solved approximately with a sub-iterative Jacobi procedure.

The two-dimensional unstructured mesh at the middle section of the fan, generated with the advancing front technique, is shown in Figure 2 together with the structured mesh used for the mapping procedure. The boundary layer region has been discretized with 12 points in the direction normal to the blade, a mesh density that is suitable for turbulent flow simulations using a wall function. The region behind the blade has been refined in order to capture the wake as accurately as possible, a feature that is important for the ability to predict the passage shock position correctly. This two-dimensional mesh is then mapped to all radial levels using the quasi-conformal mapping procedure of Section 3. Two of these sections, namely hub and tip, are shown in Figures 3 and 4 together with the corresponding structured meshes that were generated for purposes of comparison. The overall semi-structured mesh is shown in Figure 5 with a zoom view of the hub section.

To illustrate the unsuitability of unstructured meshes for such applications, a fully unstructured mesh was created for the same geometry, the suction side being shown in Figure 6. It is clearly seen that a large number of points are needed in the leading edge area in order to satisfy the resolution requirements, a feature which creates an unacceptably high overhead in the total number of points. It should also be noted that, for this particular geometry, the generation of an efficient viscous mesh will be very difficult using a totally unstructured grid because of the boundary layer considerations.

Since most turbomachinery blades are discretized using a totally structured mesh, it is appropriate to compare the performance of the semi-structured mesh against such a benchmark. An H-type structured mesh, shown in Figure 7, was generated for this purpose. This mesh contains the same number of points on the blade surface and at the outflow boundary as the semi-structured mesh, which is also shown in Figure 7. However, it has about 30 per cent more points than its semi-structured counterpart because of the superfluous

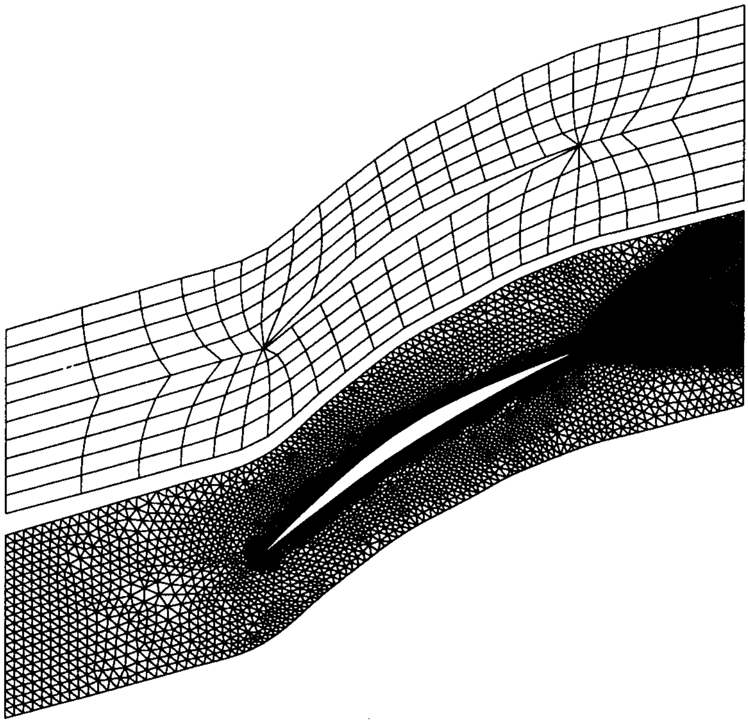


Figure 2. Unstructured and mapping mesh at middle section.

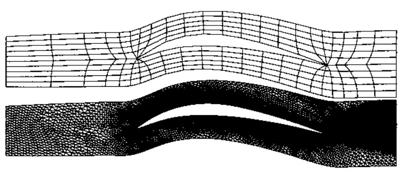


Figure 3. Unstructured and mapping mesh at hub section.

points at the inflow section. These points are needed for refining the structured grid around the blade so that the boundary layer can be resolved. Furthermore, it can be seen that the mesh refinement in the wake region is much better for the unstructured mesh. The characteristics of each grid type are listed in Table I. The steady state flow results obtained from the semi-structured mesh are compared with those obtained from the structured H-mesh, both computations being performed using the same non-linear Navier–Stokes code. Since the purpose here is to compare the relative quality of the two solutions rather than to validate the

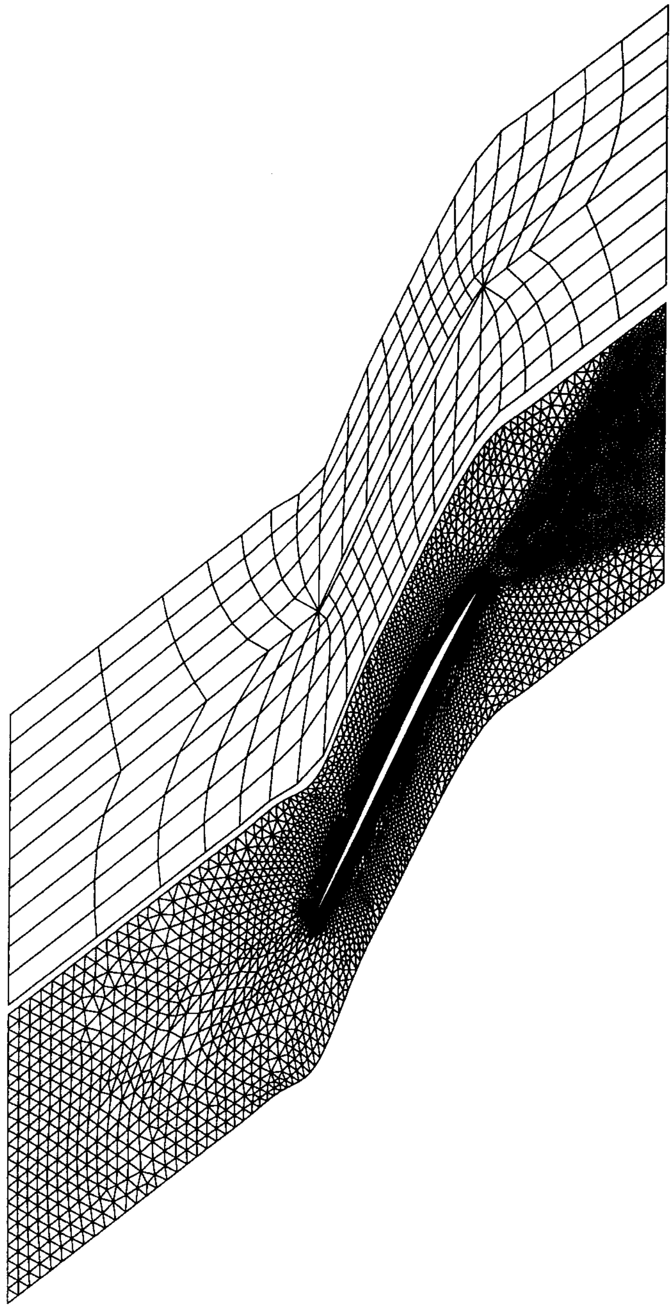
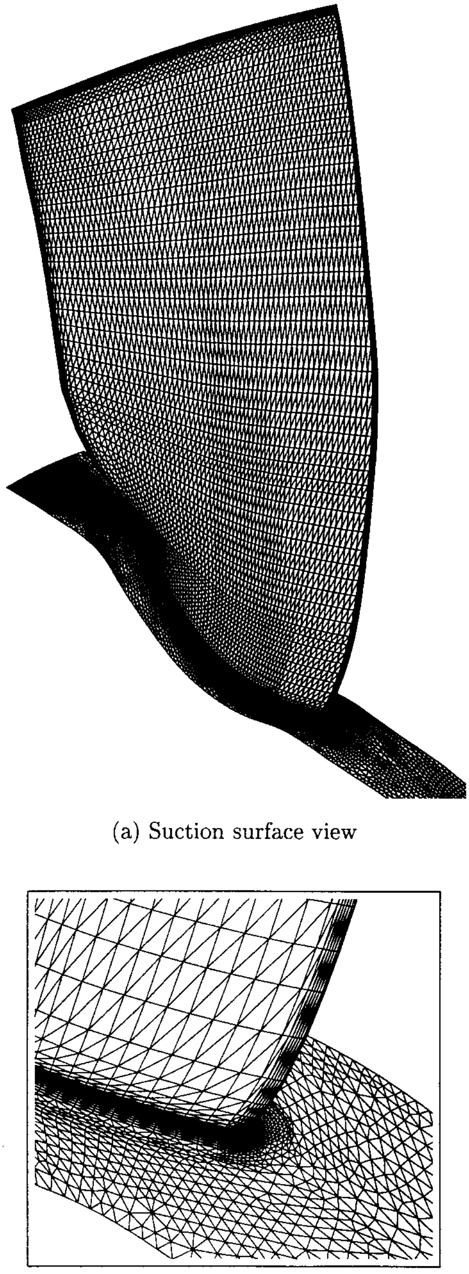


Figure 4. Unstructured and mapping mesh at tip section.



(a) Suction surface view

(b) View at hub section

Figure 5. Semi-structured mesh for a fan blade.

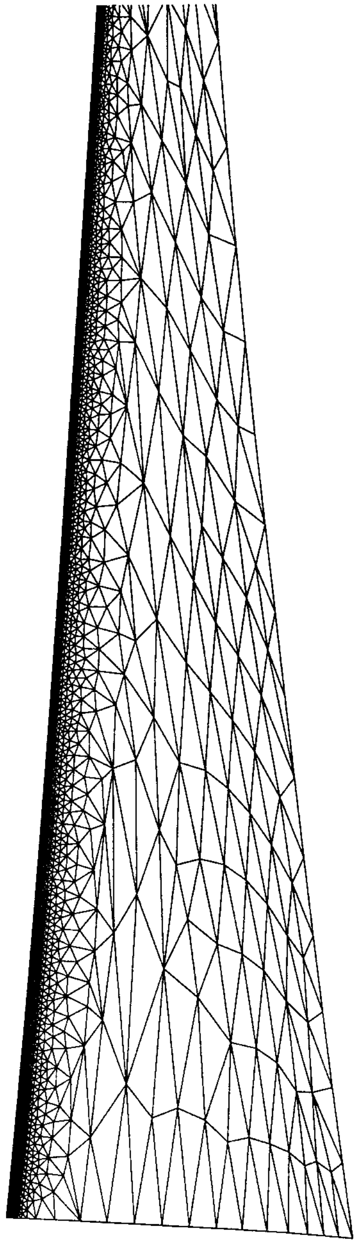


Figure 6. Fully unstructured mesh on suction surface.

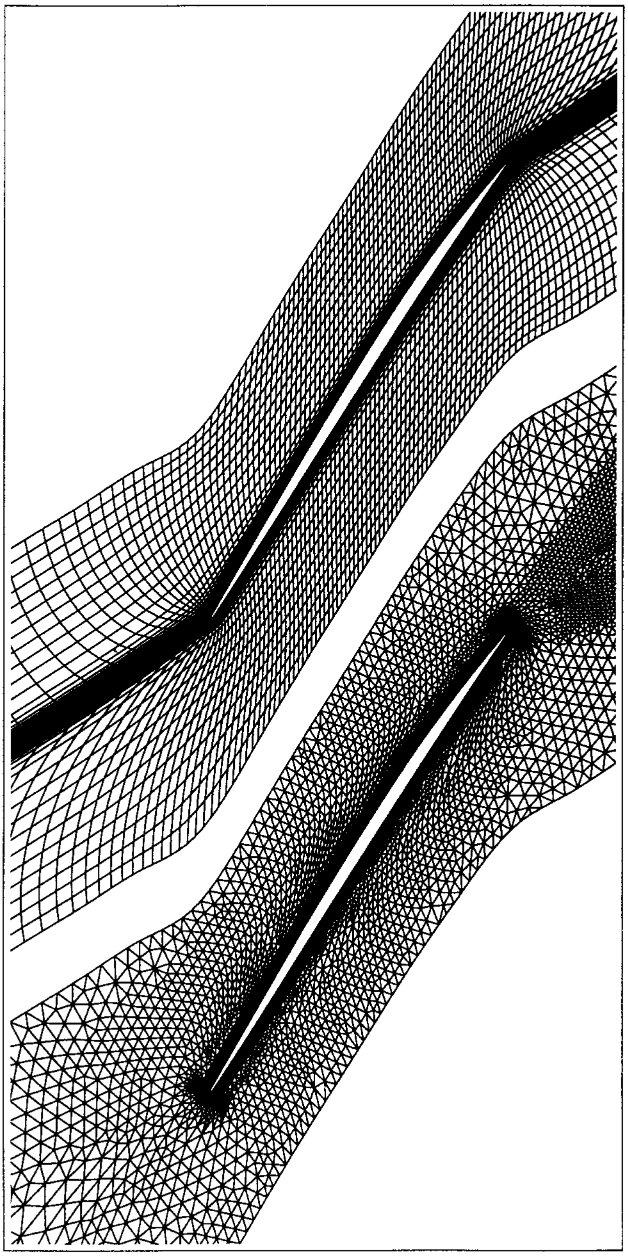


Figure 7. Structured and semi-structured meshes at 80 per cent span.

Table I. Comparison of grids for the fan blade.

Type	Flow	Number of points	Time steps to same convergence level	Normalized CPU time
Structured	Viscous	130 000	3000	153
Semi-structured	Viscous	100 000	2000	100

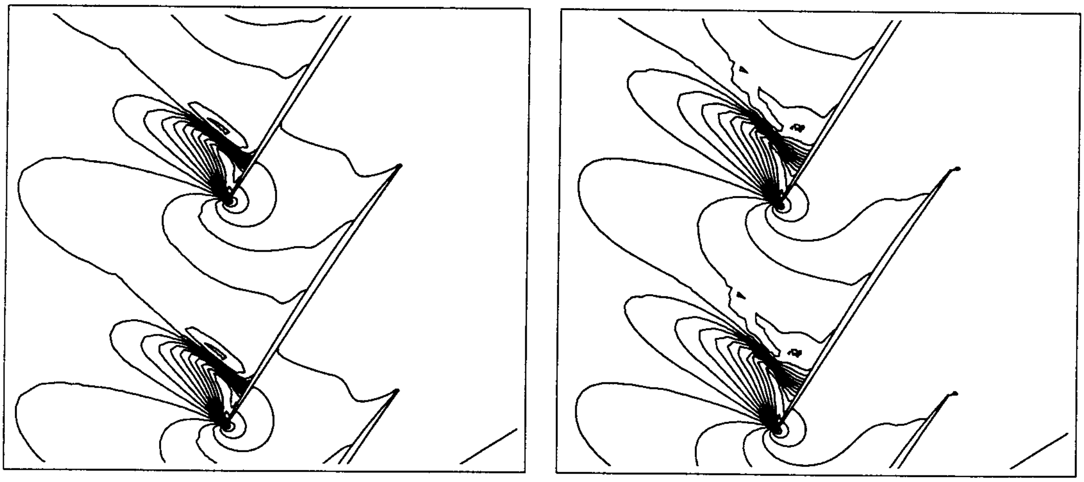
solution itself against some other code or measured data, emphasis will be placed on the general features of the two solutions that are being compared. The fully unstructured mesh was excluded from this comparison because of its inviscid discretization. It was felt that it would have been impractical to generate a fully unstructured viscous mesh using about the same number of points.

Figures 8 and 9 show the results at 80 per cent blade span. The semi-structured mesh solution is seen to have a better-defined shock. Figure 10 shows the mass flow contours at the outflow boundary of the computational domain. The general features of both solutions are similar but, as can be seen from the mass flow convergence plot of Figure 11, the convergence of the mass flow rate is much poorer for the structured mesh.

Additionally, the solutions converge significantly faster in the case of the semi-structured mesh, a feature which can be seen from the time history of the residuals plotted in Figure 12. The blip in the time history is due to a change in the artificial dissipation coefficient in an

(a) Structured mesh (b) Semi-structured mesh

Figure 8. Mach number at 80 per cent span.



(a) Structured mesh (b) Semi-structured mesh

Figure 9. Pressure at 80 per cent span.

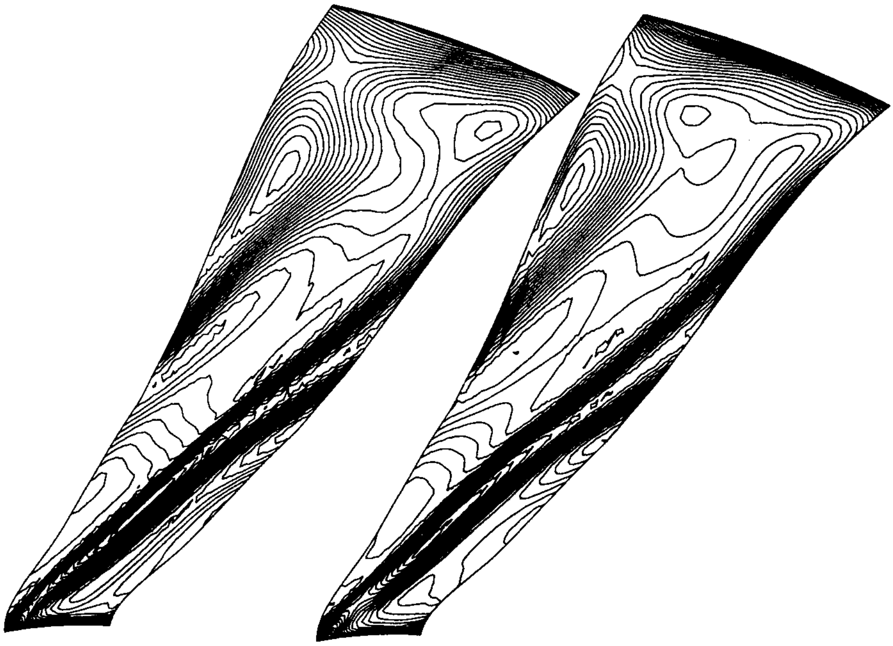


Figure 10. Mass flow contours at outflow boundary: structured (left) and semi-structured (right).

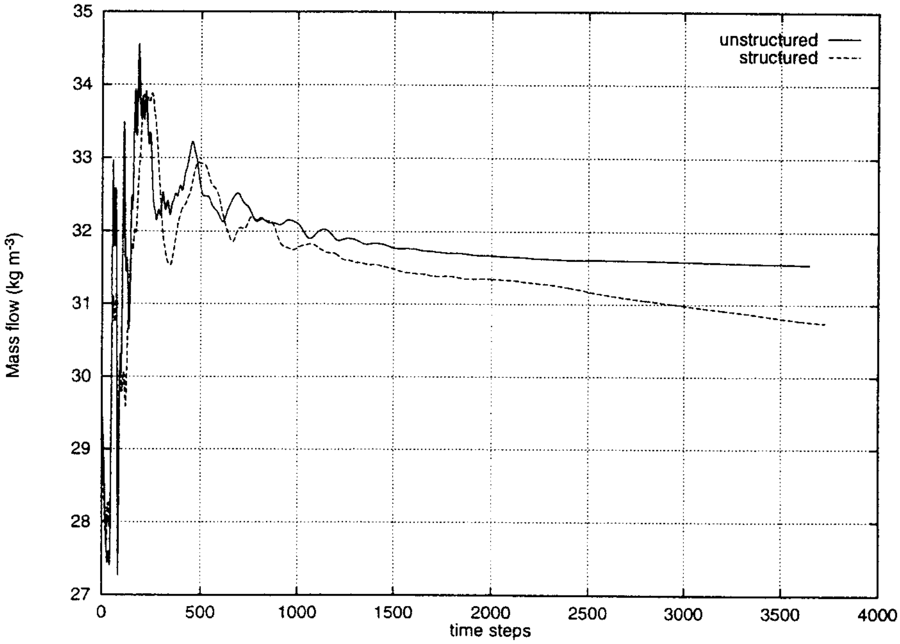


Figure 11. Mass flow time history.

attempt to improve the convergence rate for both meshes. The computing times required to obtain comparable steady state solutions are given in Table I.

5. CONCLUDING REMARKS

1. A method to generate semi-structured prismatic meshes for turbomachinery blades has been presented. The unstructured mesh in the axial and tangential directions offers more flexibility than standard structured O-type and C-type meshes, both in terms of skewness minimization and smoothness optimization.
2. Using the same solver, the solution obtained on the semi-structured mesh seems to be superior to that obtained on the corresponding structured mesh in terms of convergence rate and smoothness. Additionally, less points are required by the semi-structured mesh to provide the same level of grid resolution required to resolve the boundary layer and the wake behind the trailing edge.
3. The use of fully unstructured viscous meshes for blade-like geometries is likely to require a much larger number of grid points, and hence, there are distinct advantages in using semi-structured meshes.

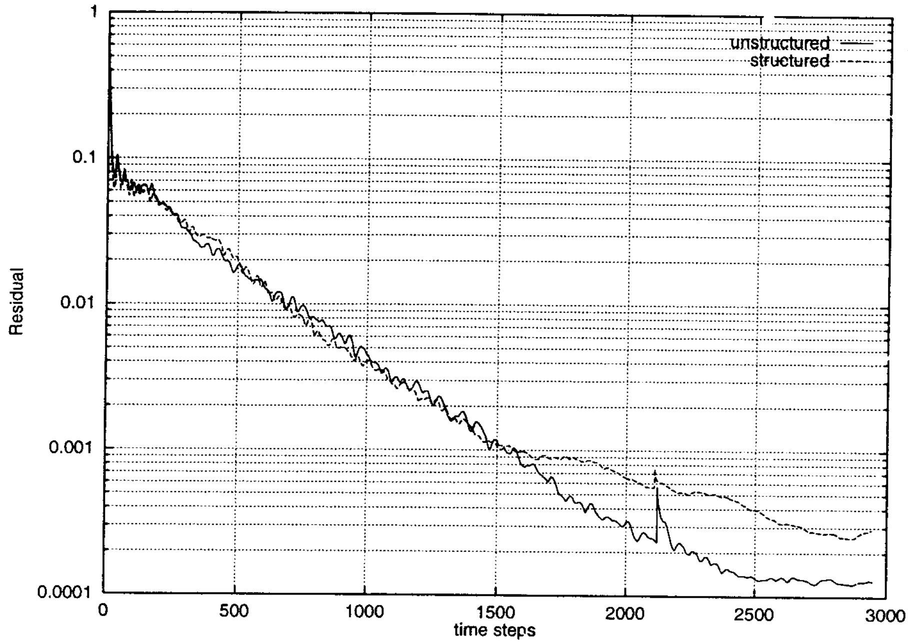


Figure 12. Residual.

4. In this work, prismatic grids are split into tetrahedra because the Navier–Stokes solver can only deal with such elements. This represents a serious limitation because of the inefficiency of tetrahedral grids for computations of the high-Reynolds number viscous flows. One of the reasons for this inefficiency is explained by counting the number of flux computations required by edge-based schemes running on tetrahedral grids. An hexahedral mesh of N points contains $3N$ sides if the boundary effects are neglected. If the same mesh is transformed by dividing each hexahedron into six tetrahedra, the number of unknowns remains the same but the number of sides becomes $7N$. If the total computational time is proportional to the number of flux computations and the flux is computed at each side, then a numerical scheme on the tetrahedral mesh is more than twice as expensive in CPU times than that on the corresponding structured grid for the same number of points.
5. Stretched tetrahedral meshes are also unsuitable for modelling the boundary layer as such situations are known to be prone to numerical problems.
6. Both the above problems can be solved by using semi-structured meshes with mixed elements. Indeed, the substitution of tetrahedral elements with hexahedral elements in the boundary layer will not only accelerate the solution, but will also make it more robust. The modification of the proposed algorithm to generate such mixed element meshes is relatively straightforward and this issue will be addressed in a forthcoming paper.

ACKNOWLEDGMENTS

The authors would like to thank Rolls-Royce plc for both sponsoring the work and allowing its publication.

REFERENCES

1. Thompson JF, Thames JC, Mastin CW. Automatic numerical generation of body-fitted curvilinear co-ordinates system for field containing any number of arbitrary two-dimensional bodies. *Journal of Computational Physics* 1974; **15**: 299–319.
2. Steger JL, Sorenson RL. Automatic mesh-point clustering near a boundary in grid generation with elliptic partial-differential equations. *Journal of Computational Physics* 1979; **33**: 405–410.
3. Brackbill JU, Saltzman JS. Adaptative zoning for singular problems in two dimensions. *Journal of Computational Physics* 1982; **46**: 442–368.
4. Morgan K, Peraire J, Peiro J. *Unstructured grid methods for compressible flows*. Technical Report 787, AGARD, 1992.
5. Mavriplis DJ. *Unstructured mesh generation and adaptivity*. Technical Report 95-26, ICASE, 1995.
6. Baker TJ. *Unstructured mesh generation by a generalized Delaunay algorithm*. Technical Report 464, AGARD, 1990.
7. Zienkiewicz OC, Morgan K. *Finite Elements and Approximation*. Wiley: New York, 1983.
8. Baldwin BS, Barth TJ. *A one-equation turbulence transport model for high-Reynolds number wall-bounded flows*. AIAA 29th Aerospace Sciences Meeting, AIAA 91-0610, January 1991.
9. Roe P. Approximate Riemann solvers, parameter vectors and difference schemes. *Journal of Computational Physics* 1981; **43**: 357–372.
10. Swanson RC, Turkel E. On central-difference and upwind schemes. *Journal of Computational Physics* 1992; **101**: 292–306.

